

Board Station for New Users Training Series

Module 1: Introduction to Board Station

Software Version 8.5_2



Copyright © 1991-1996 Mentor Graphics Corporation. All rights reserved.
Confidential. May be photocopied by licensed customers of
Mentor Graphics for internal business purposes only.

The software programs described in this document are confidential and proprietary products of Mentor Graphics Corporation (Mentor Graphics) or its licensors. No part of this document may be photocopied, reproduced or translated, or transferred, disclosed or otherwise provided to third parties, without the prior written consent of Mentor Graphics.

The document is for informational and instructional purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in the written contracts between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

RESTRICTED RIGHTS LEGEND Use, duplication, or disclosure by the Government is subject to restrictions as set forth in the subdivision (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013.

A complete list of trademark names appears in a separate "Trademark Information" document.

Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.

This is an unpublished work of Mentor Graphics Corporation.

TABLE OF CONTENTS

About This Training vii

Workbook Organization	viii
Related Documentation	viii
Documentation Conventions.....	viii
PCB Tool Function Set	x
Mentor Graphics Command Set	xi
Installation Procedure	xi

Lesson 1

Introduction..... 1-1

Course Prerequisites	1-1
Workshop Objectives	1-1
Workshop Overview	1-2
Design Process Overview	1-4

Lesson 2

The Environment 2-1

Objectives.....	2-1
The Falcon Framework	2-2
Common User Interface	2-3
Session Window	2-4
Mouse Buttons.....	2-6
Menus	2-8
Dialog Boxes	2-10
The Dialog Navigator.....	2-12
Prompt Bars	2-14
Mid-Command Freedom	2-15
Popup Command Line.....	2-16

TABLE OF CONTENTS [Continued]

Lesson 2

The Environment (Continued)

Mapping Commands to Functions 2-16

Key Definitions 2-18

Invoking PCB Tools 2-19

The Notepad Editor..... 2-22

Online Help 2-23

Online Documentation 2-24

Lab Exercise 2-27

Lab 1

The Environment2-27

Introduction 2-27

Procedure 2-27

 Invoking and Setting up the Design Manager..... 2-28

 Invoking the BOLD Browser 2-30

 Opening a Document 2-32

 Using Dialog Boxes..... 2-33

 Using Prompt Bars 2-34

 Using the BOLD Browser 2-35

 Ending the Lab Exercise 2-37

Glossary G-1

LIST OF FIGURES

The Design Manager Navigator Window	xii
Locating the Go To Button	xiii
The Add Pulldown Menu	xiii
Locating and Navigating into the Training Directory	xiv
The Source of the Training Data	xvi
The PCB Design Process	1-4
Session Window	2-5
Mouse Buttons	2-6
Scroll Bars	2-10
Radio Buttons	2-10
Push Buttons	2-11
Stepper Arrows	2-11
Check Buttons	2-11
Dialog Navigator	2-12
Prompt Bar	2-14
Stacked Prompt Bars Showing Mid-Command Freedom	2-15
Popup Command Line	2-16
Mapping Commands to Functions	2-17
Design Manager	2-20
Notepad Editor Window	2-22
Quick Help Message Box	2-23
PCB Bookcase	2-26
Location of the Window Maximize Icon	2-28

LIST OF TABLES

Key Definitions 2-18

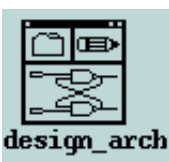
Documentation Bookcases 2-24

About This Training



Welcome to the *Board Station for New Users Training Series*. This training will help you learn to use the following PCB tools:

Design Viewpoint Editor (DVE) is used to create and modify viewpoints for design evaluation by other applications.



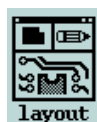
Design Architect (Schematic and Symbol Editors) is used to create and modify schematics, component symbols, and electrical properties.



LIBRARIAN (PCB Graphics Editor) is used to create and modify PCB geometry data, catalog files, and mapping files.



PACKAGE (PCB Component Package Editor) is used to assign logical symbols to physical geometries.



LAYOUT (PCB Layout Editor) is used to place geometries and route traces on PCB designs.



FabLink (PCB Layout-to-Manufacturing Editor) is used to generate manufacturing data, drawings, and reports.

This training workbook contains conceptual information and lab exercises that use the PCB tools. Instructions for installing the necessary data for the lab exercises begin in the section “Installation Procedure” on page xi.

Workbook Organization

For an overview of the organization and content of all the modules of the *Board Station for New Users Training Series*, refer to section "Workshop Overview" on page 1-2 of *Module 1: Introduction to Board Station*.

Related Documentation

For a complete listing of the manuals that make up the PCB documentation set, refer to section "Guide to the Documentation" in the *PCB Products Overview Manual*. The *PCB Products Overview Manual* describes how each manual can help you in the design process. You can find a listing of all Mentor Graphics manuals in the *Mentor Graphics Technical Publications Overview Manual*. Both of these manuals are available in INFORM.

In addition, each module of this course contains the names of the manuals that relate to the tool or tools discussed in that module.

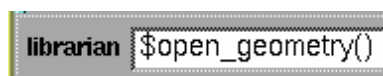
Documentation Conventions

Documentation conventions define how information such as menu paths, file names, and operating system prompts are differentiated within text. You can find a complete description of the conventions used in describing the PCB products in the Documentation Conventions section of the *PCB Products Overview Manual*.

Some important conventions for this workbook are:

Sys V>

- UNIX operating system shell prompts are represented by *Sys V>*. The shell command examples in this workbook often show UNIX (Sys V, Bourne Shell) commands.
- Command line examples without a *Sys V>* prompt are examples of functions or commands executed from within a PCB tool.



```
librarian $open_geometry()
```


- Literal file names, literal portions of file system pathnames, and literal values are shown in *italic* font. Within a literal pathname that has user-specific portions, the user-specified portions are indicated by standard font.

Sys V> **cd** *\$MGC_HOME/shared/training/board852nwp*

your_home

- The convention *your_home* is used to indicate the pathname to your home directory. When you see a pathname in this workbook that begins with *your_home*, you substitute the pathname to your home directory.

Sys V> **cd** *your_home/mgc/startup*

your_path

- The convention *your_path* is used to refer to a pathname that may be different on each user's workstation.

Sys V> **cd** *your_path/training*

location maps

- Soft prefixes, such as \$MGC_PCBPARTS and \$PROJECT_XYZ, and location maps are used to indicate pathnames to resources on the network that many users share.

After your system administrator has set up a location map and has defined the \$MGC_LOCATION_MAP environment variable, you should be able to use the soft prefixes shown in this manual.

For more information about location maps and soft prefixes, refer to the "Design Management with Location Maps" section in *Design Manager User's Manual*.

- Pulldown menu pathnames are shown in **bold** font. The right angle bracket separates the items in a menu pathname. The first name in the path (*File* in the example) is the menu name you see in the menu bar across the top of the application.

File > Save Design

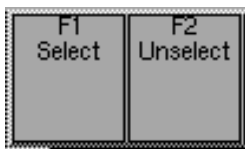
- The first item in a menu path is sometimes shown enclosed in brackets to indicate the menu is a popup menu. You access popup menus from anywhere in an edit window by pressing and holding the Menu mouse button (the right mouse button by default). The contents and name of the popup can change depending on the process you are working through. The title of the popup menu is shown in the brackets.

[Instance] Select > Select Area

- Palette pathnames are shown in **bold** font and begin with the identifier, **Palette**. If the application offers more than one palette menu, the name of the palette menu will precede the word *palette*. The right angle bracket separates the items in the pathname. For example:

Palette > Save Design

Add_Route Palette > Choose Symbol



- A function key is identified by the name of the function the key executes. For example:

Press the Unselect function key to unselect selected objects in the Edit window.

The Softkey window, located just above the Transcript window, shows the function key assignments on your workstation. In the example above, the Unselect function key is F2.

PCB Tool Function Set

While invoked on a PCB tool, you can execute a PCB tool-specific operation from the popup command line by typing a function. For each PCB function there is a unique set of argument types associated with that function. The *PCB Products Userware Function Reference Manual* documents the available functions and defines the argument types.

Mentor Graphics Command Set

Mentor Graphics also has a global command set that can be used across all Mentor Graphics applications. Some of these commands are used within an application and others are used at the operating system level. The Mentor Graphics command set is documented in the *Common User Interface Reference Manual*.

Installation Procedure

Use the following procedure to copy the lab data to your workstation. This exercise assumes you are authorized to invoke the Design Manager.

Perform the following steps:

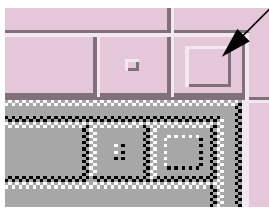
1. Log in to the workstation by entering your user name and password assigned by your system administrator or instructor.
2. Invoke the Design Manager by entering the following command at the shell prompt indicated by *Sys V>*.

```
Sys V> $MGC_HOME/bin/dmgr
```

After a moment, the Design Manager displays in a window. You use the Design Manager to create a directory in your home account that contains the training data for this training course. After you have created the directory, you locate and copy the training data into it.



\$MGC_HOME is a variable your system administrator sets to the location of your MGC tree, where the Design Manager and PCB software and training data are stored.



3. If you are working on an HP workstation, place the cursor on the session Maximize Window button in the extreme upper right corner, and click the Select mouse button. If you are working on a Sun workstation, choose the **Full Size** menu item from the window menu.

The Design Manager session fills the entire screen, and there should be a Tools window and a navigator window visible.

In the next steps, you copy the source training design data into your new training directory.

4. In the Design Manager, you should see a navigator window, similar to the partial one shown in Figure 1. Read the current directory location pathname shown at the top of the navigator window. Decide whether this location is good for adding the directory that will contain your training data. If this is a good place to add your new training directory, you can skip to step 5. If you want to change the current directory location, place the cursor on the four-way *go to* button shown in Figure 2, and click the Select mouse button (the left mouse button by default). In the prompt bar that is displayed, enter the pathname to the directory in which you want to create your new training directory.

Usually, students locate their training directory in their \$HOME directory (their home account).

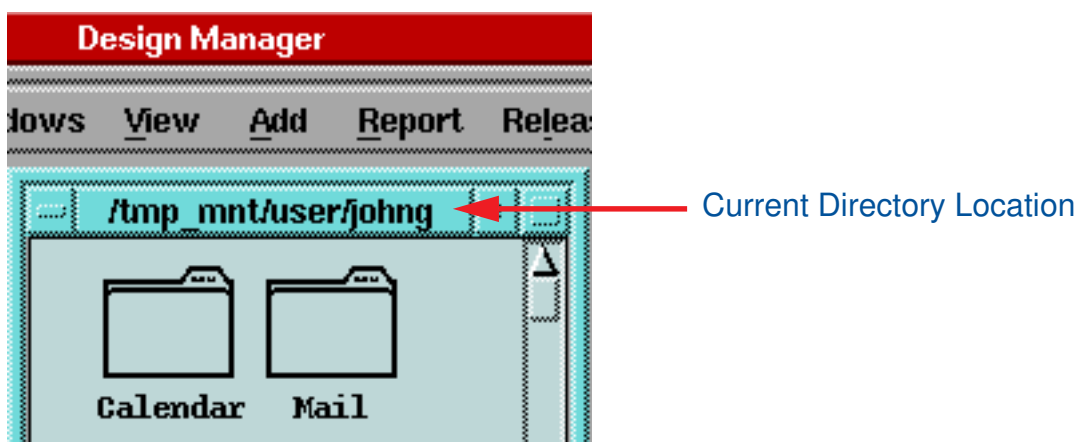


Figure 1. The Design Manager Navigator Window

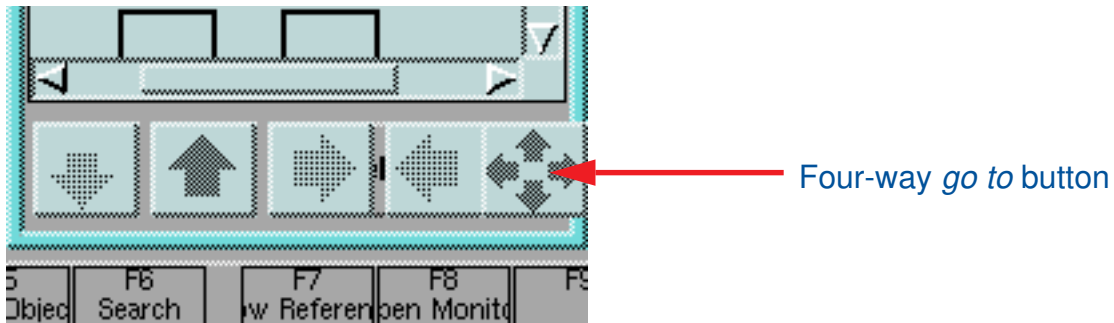


Figure 2. Locating the Go To Button

5. Check the contents of your current directory to make sure you do not already have a directory named *training*. If you have a directory named *training*, you can use it and skip to step 6. If you do not have a directory named *training*, place the cursor on the Add pulldown menu, as shown in Figure 3, and hold down the Menu mouse button (the right mouse button, by default). When the contents of the Add menu are displayed, place the cursor on the **Add > Directory** menu item and release the Menu mouse button. In the prompt bar that is displayed, enter: **training** and then either click the Select mouse button (the left mouse button by default) on **OK**, or press the Return key.

A directory named *training* is added to your current directory.

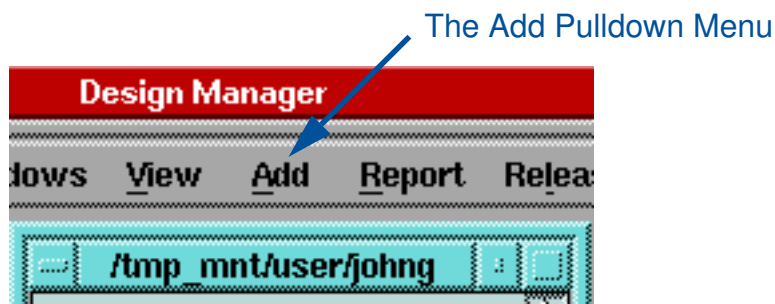


Figure 3. The Add Pulldown Menu

6. Locate the icon for your *training* directory in the navigator window, as shown in Figure 4. You might have to use the scroll bar to find it if you have many items in your current directory. After you locate the *training* directory icon, place the cursor on it and click the Select mouse button. Finally, press the Explore Contents button by placing the cursor on it and clicking the Select mouse button.

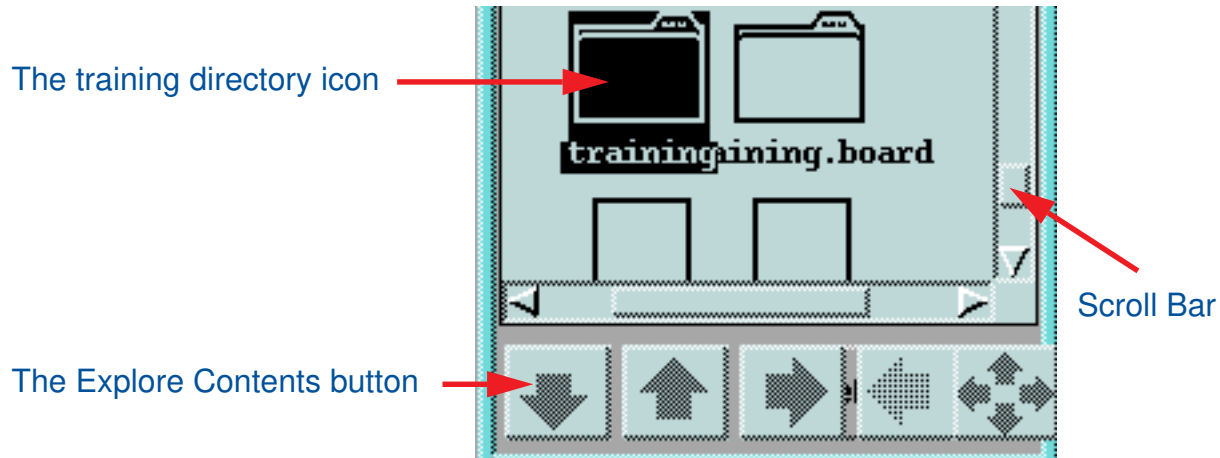


Figure 4. Locating and Navigating into the Training Directory

Next you create a directory named *board_new* within the training directory. You place all your Board Station training data in the *board_new* directory.



You can store training data for other Mentor Graphics courses under this training directory, as well. The board_new subdirectory is where you copy the training data for the Board Station for New Users Training Series. You can create additional subdirectories under the training directory to hold data for additional Mentor Graphics courses.

7. Choose the **Add > Directory** menu item. In the prompt bar, enter **board_new**, and **OK** the prompt bar.

A *board_new* directory/icon should now be visible in the *training* directory.

8. Select the *board_new* icon by placing the cursor on it and clicking the Select mouse button. Next, navigate to the *board_new* directory by clicking on the Explore Contents (down arrow) button.

Next you create another navigator window and locate the source of the training data for this course. After you locate the source of the training data, you copy it to your *board_new* directory.

9. Choose the **Windows > Open Navigator** menu item. When the new navigator window is displayed, click on its Go To (four-way arrow) button. In the dialog box that is displayed, enter: **\$MGC_HOME/shared/training**, then **OK** the dialog box by either clicking on **OK**, or by pressing the Return key.

The contents of the new navigator window should show all the available training data packages you have at your site.

10. In the new navigator window, locate the *board852nwp* icon, and select it so it appears as shown in Figure 5 (your *board852nwp* icon might have an arrow in it to indicate it is a link). Click the Explore Contents (down arrow) button to see the contents of the training data.

If you do not see a *board852nwp* icon, it has not been installed at your site. Please have your system administrator install this training package before continuing with the training.

When you open the *board852nwp* directory, you should see directories named *mod2*, *mod3*, *mod4*, and so on through *mod8*. You need to copy the data appropriate for the module you are currently working with. For example, if you are working on Module 3: Creating PCB Design Geometries, you need to copy the training data *mod3*. Module 1: Introduction to Board Station requires no training data.



Figure 5. The Source of the Training Data

11. From the list of module design data (*mod2*, *mod3*, and so on), select the icon for the module you are currently working on. For example, if you are working on Module 3, select the icon *mod3*.

In the next step, you copy the selected directory/icon to your *your_path/training/board_new* directory.

12. After you select the icon you want, place the cursor on it, hold down the Shift key, and also hold down the Select mouse button (the left mouse button by default). While holding down the Shift and Select mouse button, move the cursor. You should see a ghost-image of the icon move with the cursor. Move the cursor/ghost-image to the other navigator window, which is your *board_new* directory. When the ghost-image is in your *board_new* navigator window, release the Select mouse button and the Shift key.

The module data you selected is now being copied to your *board_new* directory. The copy process may take up to several minutes.

You have completed copying data. If you want to copy data for other modules you can do so now.



*Pay attention to the amount of free disk space you have where your *board_new* directory is located. The data for each module is between 10 and 15 Megabytes. If you do not have enough space for more than one module's data at a time, you can delete each data directory when you finish using it before copying the data for the next module.*

Lesson 1

Introduction

Welcome to the Board Station for New Users Training Series. This training is designed to introduce you to the process of designing printed circuit boards using Mentor Graphics Board Station tools.

This module presents an overview of the training and a brief description of the process of circuit board design.

The lab material presents sequential tasks that demonstrate the conceptual topics. Some tasks include specific step-by-step instructions, while other more common tasks do not. Although most lab material consists of text, illustrations are included when discussing specific board design elements.

Course Prerequisites

In order to benefit sufficiently from the materials presented in this training series, you need experience in the art and technology of designing printed circuit boards. This training series does not cover any of the basic techniques and processes that are necessary to design printed circuit boards. This training series concentrates on the use of Design Architect and Board Station tools to improve the quality of your board designs.

Workshop Objectives

The *Board Station for New Users Training Series* is designed to introduce you to the basic processes and techniques of creating circuit boards with Board Station tools. This training is designed to give you a beginning experience and orientation to the Board Station design process.

Workshop Overview

The *Board Station for New Users Training Series* is not a course in printed circuit board design. Its intent is to develop your experience using Common User Interface elements and V8 Board Station tools and design processes. The knowledge you gain in this course increments your existing knowledge and experience in circuit board design.

To gain the most from this training, you should know how to perform the basic steps of designing a printed circuit board. You should have some knowledge of the basic circuit board technologies used at your company. You should also know the standard design rules that are required for your design and manufacturing processes.

The *Board Station for New Users Training Series* consists of seven training modules. If you are taking this training as a self-paced personal learning program, you can complete any module in any order because the information and lab training data is self-contained for every module. If you are taking this training in an instructor-led workshop, modules 1 through 4 are covered the first week, and modules 5 through 7 are covered in the second week of class. The training modules roughly follow the entire design process for designing and creating a PCB. The modules are:

- Module 1: Introduction to Board Station

A brief overview of Board Station and design processes, the training materials, the user interface, the Design Manager, and the Bold Browser.

- Module 2: Preparing a Design for PCB

An introduction to Design Architect with PCB personality module. You learn how to view and edit a design schematic, how to add symbols, wires, and design properties, and how to extract information about the design. You also learn how to create specialized design viewpoints for PCB and create back annotation objects.

- **Module 3: Creating PCB Design Geometries**

This module covers parts libraries, creating and editing pin, via, component, board, and generic geometries, creating and editing catalogs, and mapping symbols to component geometries.

- **Module 4: Packaging a Design for LAYOUT**

Packaging concepts and the input to the PACKAGE tool, control of flow of properties into PACKAGE, control how logic is packaged, interactive and automatic packaging, back annotation to the schematic, and PACKAGE output are covered in this module.

- **Module 5: Placing Components on a Circuit Board**

In this module, you learn how to control and set up placement using properties. You learn how to both interactively and automatically place components on a board, and you learn how to evaluate and adjust placement.

- **Module 6: Routing Traces on a Circuit Board**

This module covers design rules, interactive routing, grid-based automatic routing, routing with the Dynamic Editor, shape-based automatic routing, and area fills.

- **Module 7: Creating Manufacturing Data**

Creating manufacturing data includes changing reference designators, creating an artwork order, creating and editing an aperture table, creating artwork and power fill areas for split power planes, creating thermal reliefs, simulating artwork data, creating test coupons, thieving patterns, and panels, and creating drill tables, drill data, and milling paths. At the end of this module, you create both fabrication and assembly drawings. You also create aperture table and drill table reports, and a bill of materials.

Design Process Overview

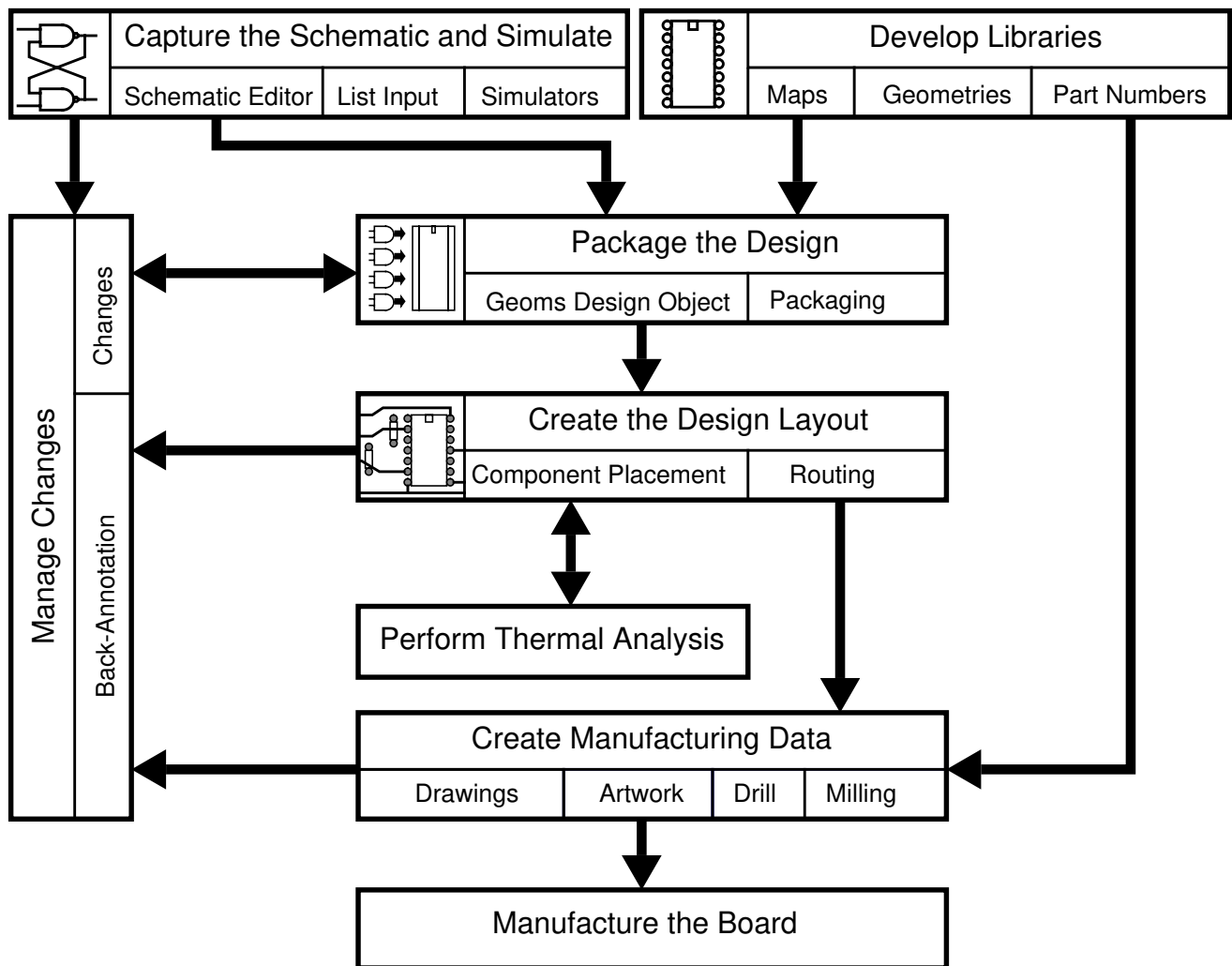


Figure 1-1. The PCB Design Process

- The design information captured by the Schematic Editor and simulated to verify functionality is processed into a design database. Input to the circuit board design tools might also be a net list in ASCII file format and a component list in ASCII file format.
- Geometry data from vendor specification sheets or engineering drawings is the source of data to develop library geometries. LIBRARIAN is a general purpose graphics editor you use to create geometries, maps, and part numbers.
- The process of packaging assigns the logical symbols from the schematic to the physical parts that are placed on the printed circuit board. In this step, the geometry information is compiled into a design geometry database. PCB PACKAGE performs these two functions.
- Printed circuit board design, the placement of geometries and the routing of traces, is performed with the LAYOUT tool.
- Thermal analysis can be performed to predict temperatures on the operating circuit board after the component placement has been determined. If the predicted temperatures are outside the tolerance limits, the component placement can be changed.
- The FabLink tool creates the data to drive a photoplotter, a drill machine, and a milling machine. You can also use FabLink to create drawings and other reports used in manufacturing.
- A design cycle is not accurately represented without accounting for the changes that occur. These changes might be caused by engineering decisions or by new information like reference designators added late in the design process.

Lesson 2

The Environment

In this module, you examine the features of the Falcon Framework. The Falcon Framework is the environment common to all Mentor Graphics tools. The Falcon Framework consists of the following tools:

- The Common User Interface, the interface for all applications.
- The Design Manager, used to invoke Mentor Graphics tools and manage design data.
- The Notepad, a text editor.
- The BOLD Browser, an application that displays online documentation.
- AMPLE, an application programming language.

Objectives

The purpose of this module is to introduce you to the most commonly used features of the Falcon Framework and the Common User Interface. You learn how to use the mouse to make menu selections and execute commands. You invoke tools using the Design Manager. You also learn how to access online manuals and other online helps. You need to become familiar with these tools so that you can feel comfortable using the other Board Station applications in later labs.

The Falcon Framework

The Falcon Framework is a powerful common environment in which all Mentor Graphics and Open Door applications run. These applications use the Falcon Framework to provide a common user interface, text editor, electronic mail system, and decision support system. The Falcon Framework consists of the following:

- Common User Interface—the user interface common to all applications; supports Open Software Foundation (OSF)/Motif standards for the look and feel of all aspects of the user interface.
- The Design Manager—an application you use to invoke tools and manage design data configurations.
- Notepad—a powerful text editor that operates in a window of any application session.
- BOLD Browser—an application that displays online documentation and help information.
- AMPLE—a powerful extension language supporting customization and integration throughout the Falcon Framework and all Falcon-based applications.

Common User Interface

All Mentor Graphics products use the Common User Interface. Because the user interface is common to all applications, Mentor Graphics products are easy to learn and use. The Common User Interface is based on OSF/Motif standards. The OSF/Motif standards are an industry-wide set of guidelines for the look and feel of the user interface.

Some important features of the Common User Interface are:

- Consistent appearance and functionality across all Mentor Graphics products.
- Multiple windows on the display screen, allowing you to access different parts of an application or different applications during a single session.
- Standard menus for performing generic tasks and accessing online help.
- Highlighting that indicates which window can receive input.
- Control buttons and menu items that appear raised and three-dimensional when activated.
- Consistent controls and actions for all types of menus.
- User control of a cursor within a window.
- Graphical controls that permit you to interact with applications.
- Access to the Common User Interface through graphics input devices, keyboard mnemonics, and the keyboard.
- Transcript and messages that provide you with information from applications.

Session Window

All V8 tools run within a UNIX shell. V8 tool windows have these features:

- Session windows are created when you invoke a tool. The session runs in an UNIX shell so you can pop between Session windows. A Design Architect Session window is shown in Figure 2-1.
- Subwindows contain tool-specific graphics and text, such as Edit windows, function keys, menus, and transcripts.
- Buttons help you manage the windows by resizing, moving, and turning them into icons.
- Highlighted window borders designate an active window (one that is ready to receive input).
- The menu bar incorporates pulldown menus.
- Multiple shell and PCB tool session windows can be available as active windows or as icons.

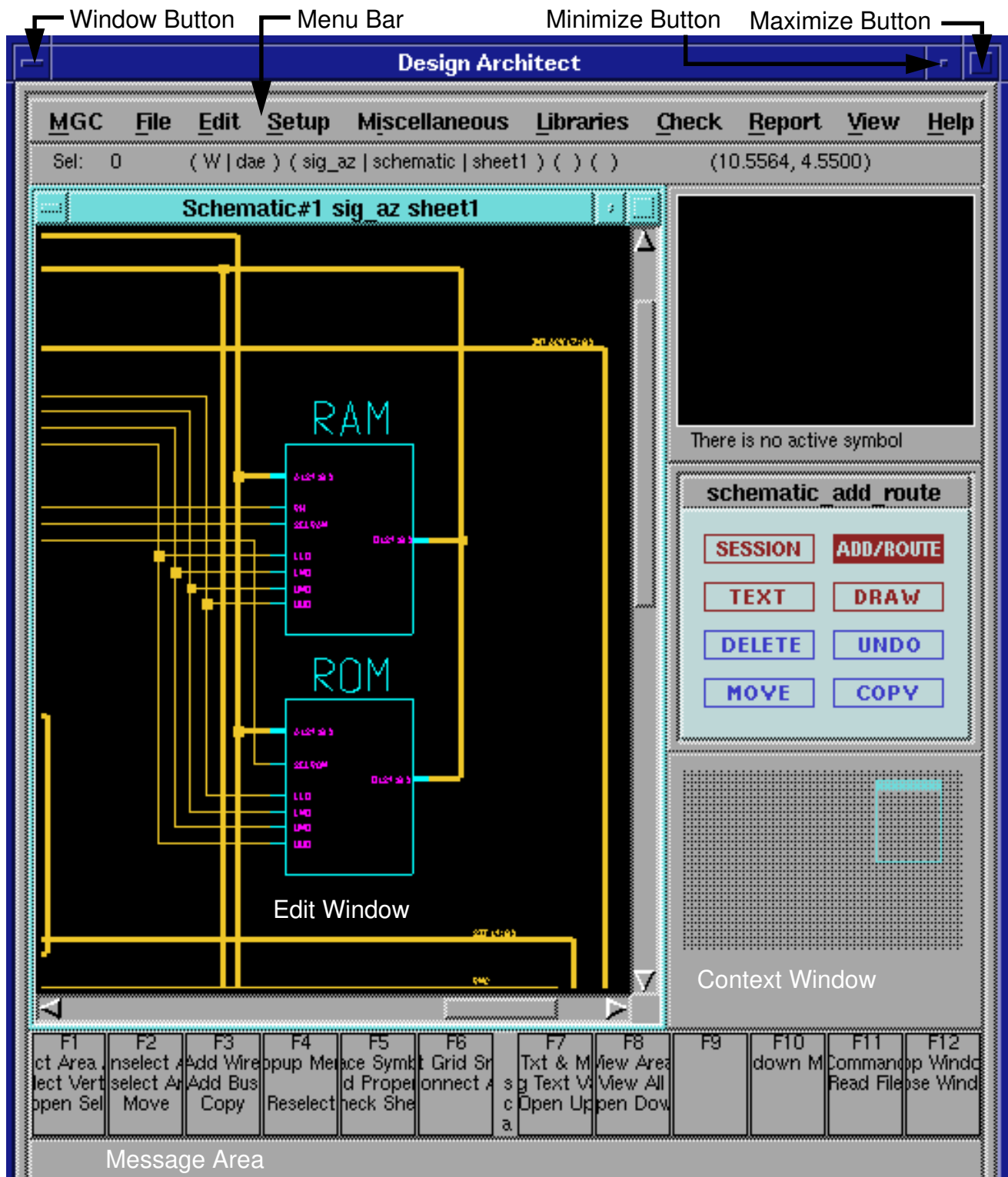


Figure 2-1. Session Window

Mouse Buttons

Mouse buttons are defined to perform specific predefined functions, or you can customize the button functions. Each mouse button can operate in a normal, shift, or control mode. Mouse buttons and their functions are shown in Figure 2-2.

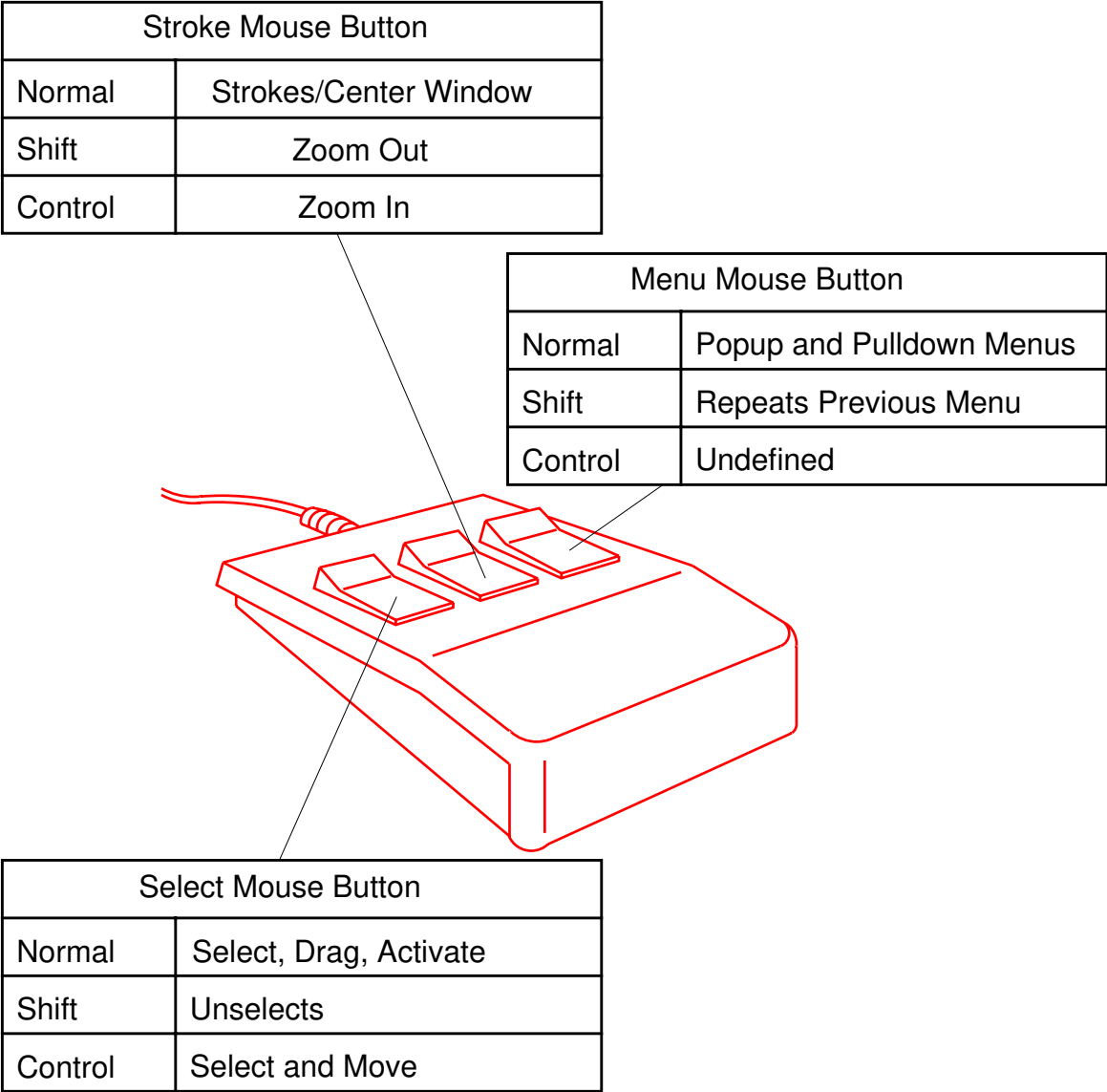


Figure 2-2. Mouse Buttons

The following can be helpful as you learn to use a mouse:

- Selections are made by pointing and clicking on a single object or dragging to encompass multiple objects.
- Board Station applications use selection and unselection filters. You specify the type of object or objects to select. For example, you can specify that only components or vertices can be chosen for selection. You also specify whether objects inside, outside, or crossing the selection rectangle can be selected.
- Double-clicking can restore an icon to a subwindow or execute a close on the Session window button.

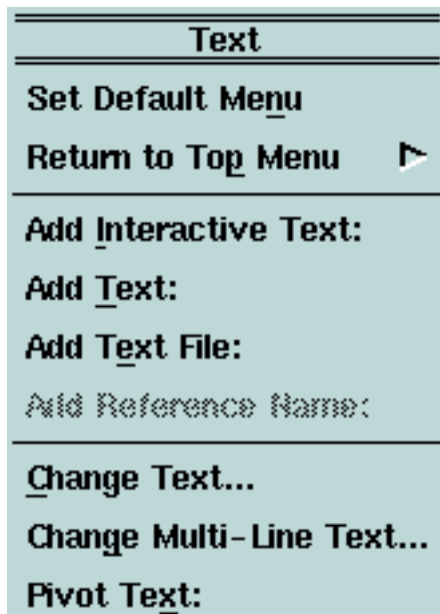
Menus

A *menu* is a graphical aid that displays a list of items representing tasks pertinent to an application. When you select a menu item, the application performs the specified action or asks for additional information before executing the action. There are four types of menus:

Session	
<u>H</u> elp	7
<u>N</u> otepad	7
<u>C</u> leanup Windows	
<u>U</u> serware	7
<u>T</u> ranscript	7
<u>S</u> etup	7
<u>D</u> esign Management	7
<u>L</u> ocation Map	7
<u>E</u> xport Screen...	

- **Pulldown Menu**—displays from the menu bar and contains menu items that represent tasks you can perform in the active window.
- **Popup Menu**—displays in the active window. It contains menu items that represent tasks you can perform in that window. The popup menu is invisible until you explicitly display it, at which time it literally pops up on the display screen. The Routing popup menu and Extended Routing submenu are shown on this page and the next page.
- **Session Menu**—displays in the session window. The session window is where the edit windows of an application display. To access the session menu, place the cursor in an area of the session window that is not covered with an edit window (you might have to change the size of an edit window to uncover some session window space), and press the Menu mouse button.
- **Extended Menu**—displays in the active window for certain categories of menu items. Pressing the SHIFT key and Stroke mouse button at the same time accesses a special type of popup menu. The extended menu displays less frequently used commands. It is also available through the standard Popup menu.

The Select or Menu mouse buttons access the pulldown menus in the menu bar. The Menu mouse button accesses popup menus in the Edit subwindow.



The following menu graphic indicators are used in menus:

- **Menu cascade arrow**—an arrow indicating that a cascading menu is present.
- **Prompt bar indicator**—a colon indicating that the selected menu item displays a prompt bar for additional command or function arguments.
- **Dialog box indicator**—an ellipsis indicating that the selected menu item displays a dialog box for additional information to the system.

Dialog Boxes

Dialog boxes are used by certain menu selections to gather additional information. Dialog boxes vary in appearance and use different types of selection buttons.

Controls provide access to dialog box items:

- Scroll bars reveal parts of lists.

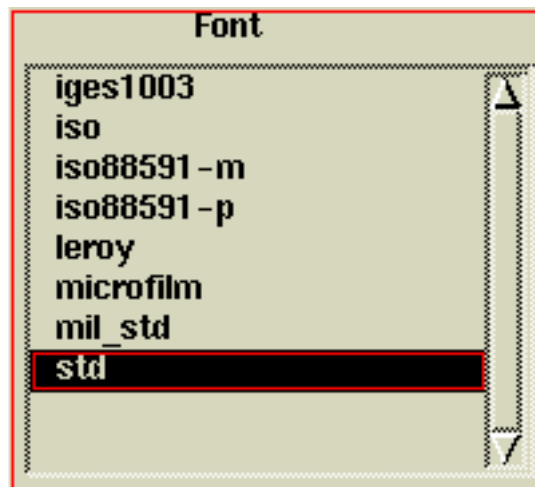


Figure 2-3. Scroll Bars

- Radio buttons allow only one selection at a time.

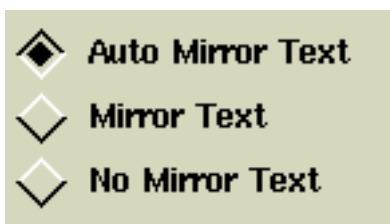


Figure 2-4. Radio Buttons

- Push buttons are labeled and allow more than one selection at a time.

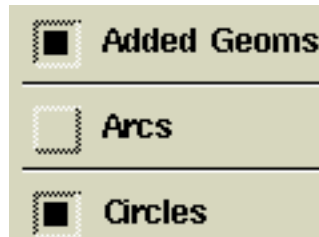


Figure 2-5. Push Buttons

- Stepper arrows reveal one optional argument at a time.



Figure 2-6. Stepper Arrows

- Check buttons allow you to select or decline options.



Figure 2-7. Check Buttons

The Dialog Navigator

The dialog navigator, illustrated in Figure 2-8, is a dialog box control you use to navigate through your designs and select design objects on which to work. Occasionally, your application might need to locate a design object on which to perform a task, such as *open* or *copy*. If your application features the dialog navigator, then you need not supply the pathname to that design object. Instead, you can use the dialog navigator to find and select the design object. For more information about design objects and references, refer to the *Design Manager Users Manual*.

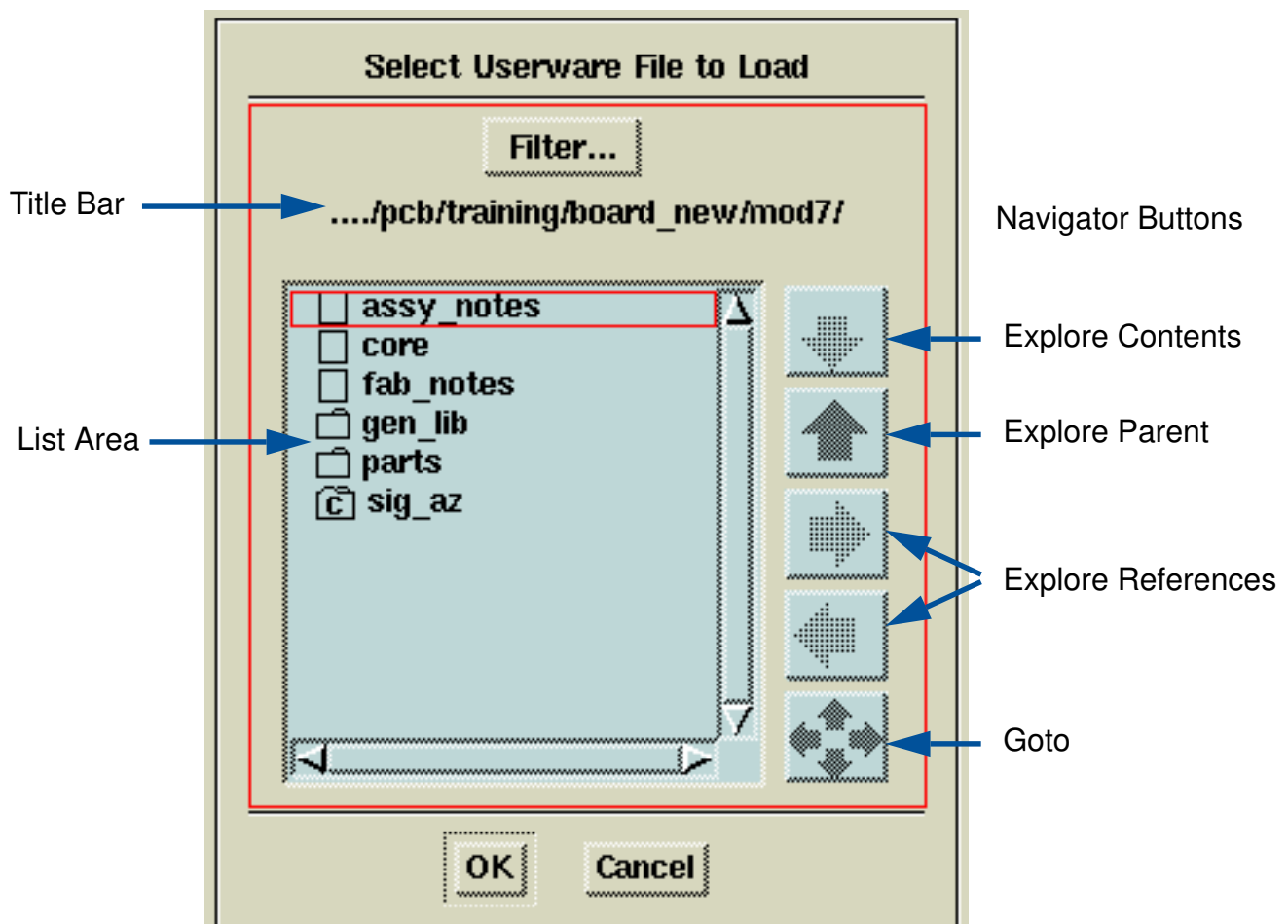


Figure 2-8. Dialog Navigator

The dialog navigator has the following features:

- A title bar, which indicates the current location.
- A list area, which displays, in alphabetic order, a list of files, directories, and design objects, with an icon preceding the name of each object.
- Navigation buttons, which allow you to navigate the file system or navigate references. These buttons have the following meaning:

Explore Contents—navigates into the selected directory and displays the contents of the directory.

Explore Parent—navigates up one directory or reference level.

Explore References—replaces the current display with the references of the selected design object.

Go To—displays a dialog box into which you can enter a file system destination directory name. After you enter the destination directory, the dialog navigator navigates to and displays the contents of that directory.

Prompt Bars

Single-line dialog boxes that request input for a few arguments appear as prompt bars. Prompt bars are displayed either by selecting some menu items or by simply typing the name of a function or command without listing the required arguments. In Board Station, prompt bars appear as a cue whenever dynamic cursor response is required. The Add Text prompt bar is displayed in Figure 2-9.

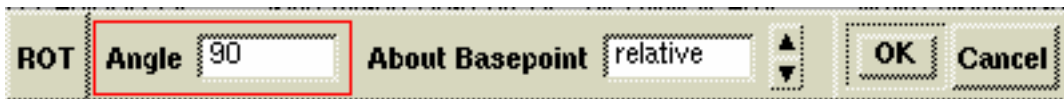


Figure 2-9. Prompt Bar

Do the following to move around in or to execute the prompt bar:

- Use the Tab key to scroll to the next field.
- Use Shift-Tab to scroll to the previous field.
- Execute a prompt bar by placing the cursor on the OK button and clicking the Select mouse button, or by pressing the Return key.
- Cancel a prompt by placing the cursor on the Cancel button and clicking the Select mouse button, or by pressing the Esc (Escape) key.

Mid-Command Freedom

You do not have to respond to a prompt bar immediately. Prompt bars stack on top of each other until you respond. In this way, while mid-way through satisfying the prompts in one prompt bar, you can, for example, change the view in an edit window or perform another function, and then later complete the original prompt bar. This ability to do a different task while part way through another task is called *Mid-Command Freedom*. Figure 2-10 shows an example of Mid-Command Freedom. In this example, the Move prompt bar, which is on the bottom of the prompt bar stack, required a location. The user took advantage of Mid-Command Freedom to choose a menu item that provided a way to specify the location that the Move prompt bar required.

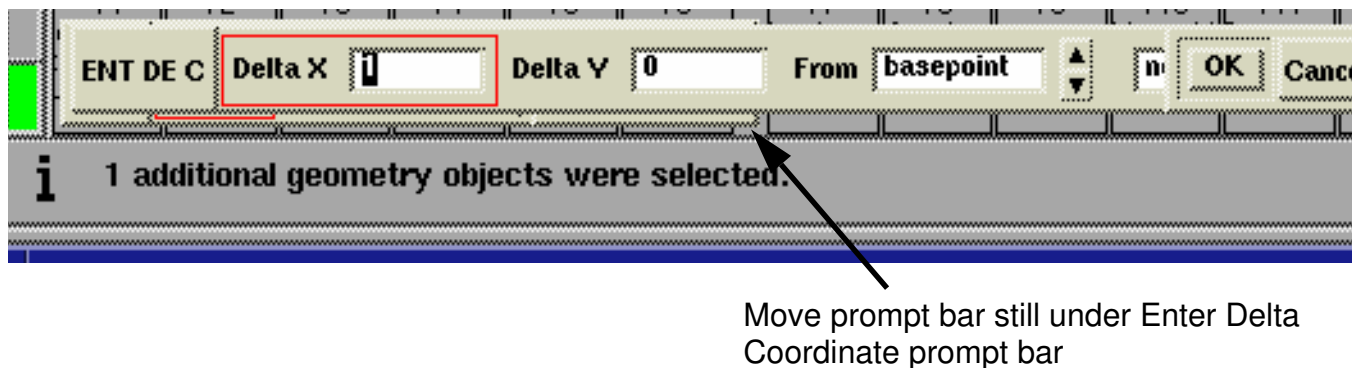


Figure 2-10. Stacked Prompt Bars Showing Mid-Command Freedom

Popup Command Line

A popup command line, as shown in Figure 2-11, automatically appears at the cursor location when you begin typing, if there is no dialog box displayed or active field in a displayed prompt bar. The title of the command line is the active window or area. You can use a popup command line to enter commands or functions directly, in addition to choosing menu items or using function keys.



Figure 2-11. Popup Command Line

To display a list of all available commands, type * **Ctrl Shift?** (type the *, then hold down the Ctrl key, the Shift key, and click the ? key). If you know one or more letters of a command, you can enter those, followed by Ctrl Shift?, and a list of all commands that start with those letters will be displayed. Do not press the Return key after entering either of these commands.

Mapping Commands to Functions

Functions are the design-tool instructions. Mentor Graphics tools use a set of commands that are mapped (registered) to the functions. You can register your own commands in the user interface using AMPLE commands.

Mentor Graphics uses a registered set of *minimum typing* characters to represent a command or function, such as **setedl** for \$set_edit_layer(). The minimum typing characters are determined by a 3-2-1-1 rule. You can enter a command by typing 3 characters from the first word, 2 characters from the second word, and 1 character from the third and fourth words of the command.

The following examples of command-line entry are equivalent:

Function syntax: `$set_edit_layer()`

Command syntax: `set edit layer`

Minimum typing syntax: `setedl`

Figure 2-12 illustrates a menu item and a command registered to a function which, in turn, calls a builtin function that defines the action performed. Palette menu items and strokes map to functions the same way as menu items do.

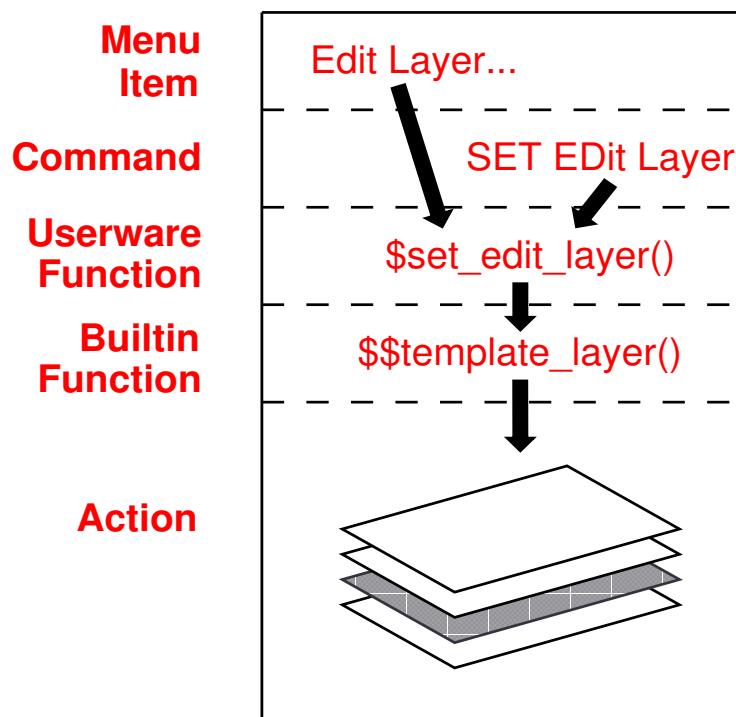


Figure 2-12. Mapping Commands to Functions

Key Definitions

Some of the most important key definitions are shown in the following table.

Table 2-1. Key Definitions

Key	Definition
Backspace	Removes the previous location point entered with the mouse (to define a path, for instance), in addition to removing the previous character.
Ctrl-U	Updates graphics. Ctrl-R also updates graphics.
Esc	Cancels prompt bars and dialog boxes.
Return	Executes prompt bars and dialog boxes, in addition to commands and functions that you type.
Tab	Moves the input position to the next field in the prompt bar or dialog box.
Shift-Tab	Moves the input position to the previous field in the prompt bar or dialog box.

You can define a key using a `$key_keyname()` function:

```
function $key_keyname() { $function(); }
```

Example: Define function key F4 to be a delete key:

```
function $key_f4() { $delete(); }
```

When you define a function key, you overwrite any pre-existing function it might have.

Invoking PCB Tools

The Design Manager, shown in Figure 2-13, offers two methods for invoking PCB tools. One method is a data-centered invocation process and the other is a tool-centered invocation process. A data-centered invocation means that in the navigator you select and open the icon that represents your design data. A tool-centered invocation means that in the Tool window you select and open the icon that represents a specific PCB tool. The initial step in invoking from the Design Manager is to open the Design Manager from an Aegis shell or UNIX window by typing:

```
$MGC_HOME/bin/dmgr
```

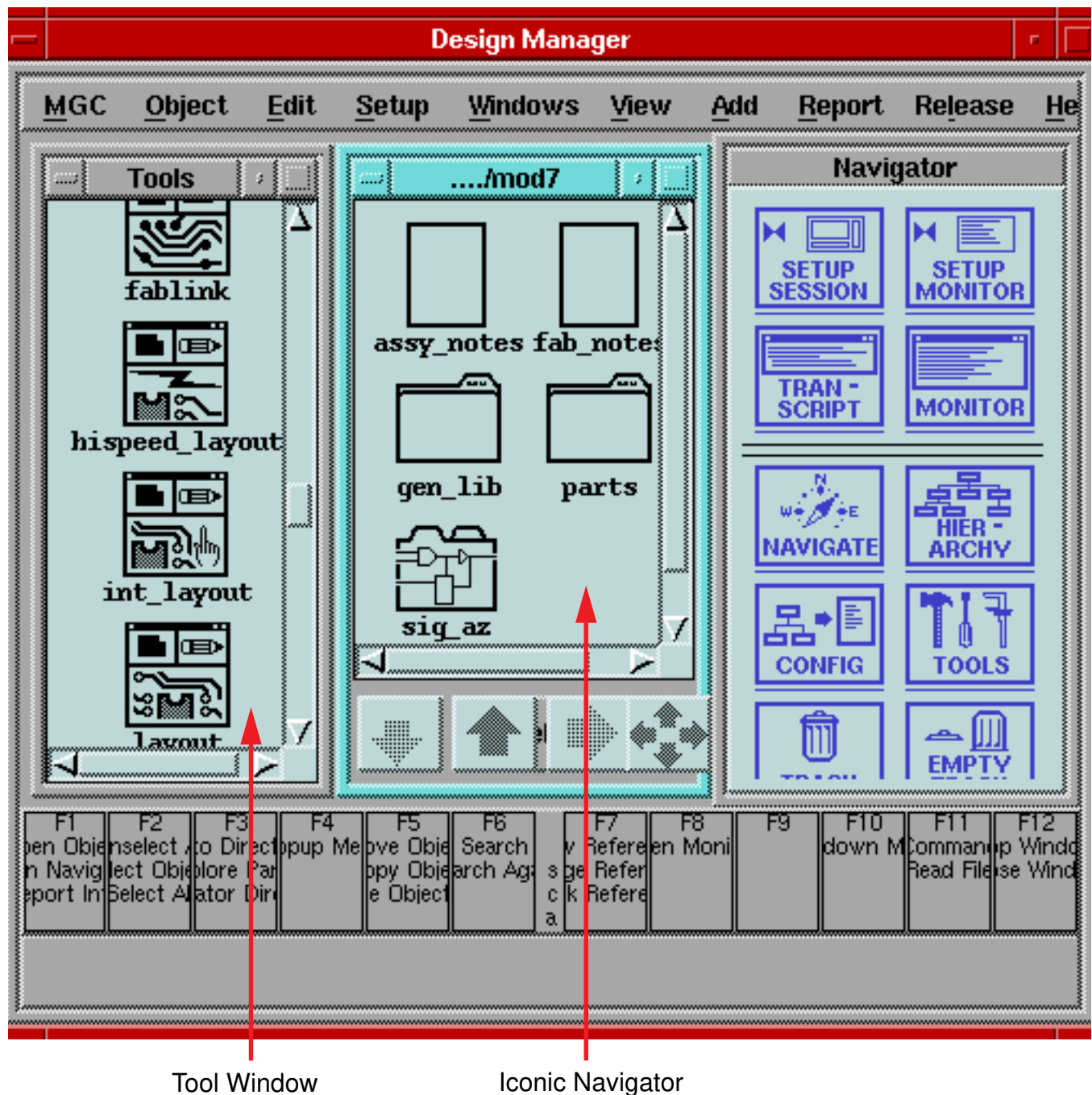


Figure 2-13. Design Manager

From the Object pulldown menu you can open the Iconic Navigator or Tool window. The navigator displays the contents of your working directory.

To invoke a PCB tool using the data-centered method:

1. Select the *pcb* design object icon representing your design.
2. Press the Menu mouse button to display the navigator's popup menu. Cascade the Open menu item to see a list of tools that can open the design folder.
3. Select the menu item that corresponds with a tool that you want to invoke, such as **Open > FabLink**.

To invoke a PCB tool using the tool-centered method:

4. Select the icon of the PCB tool that you want to invoke.
5. Press the Menu mouse button to display the Tool window menu and select the menu item **Open**.
6. Select the folder representing your design data in the dialog navigator list box that appears and press the OK button at the bottom of the list box.

The Notepad Editor

The Notepad Editor is a full text editor window that is available in the Falcon Framework Environment. Text can be cut and pasted between windows similar to other text editors. Because the reports generated by Board Station tools are displayed in Notepad windows, you can use the text editing features between almost any windows during a design session.

You can also access the Notepad editor from the menu bar during any session using Mentor Graphics tools. A Notepad window is shown in Figure 2-14.

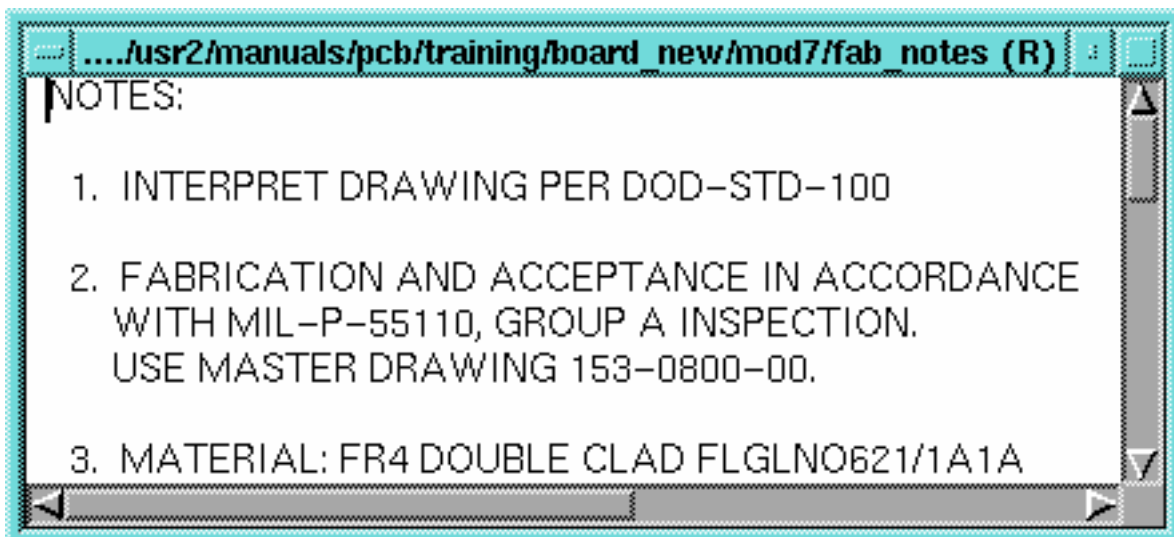


Figure 2-14. Notepad Editor Window

Online Help

All Mentor Graphics tools enable users to quickly get help on functions, either in the form of brief descriptions called *Quick Help* or as online reference pages.

The two types of online help are available in different formats:

- Quick help is displayed in a message box, as shown in Figure 2-15.
- A reference page from an appropriate reference manual is displayed online.

Quick help can be accessed from a dialog box by choosing the Help button.

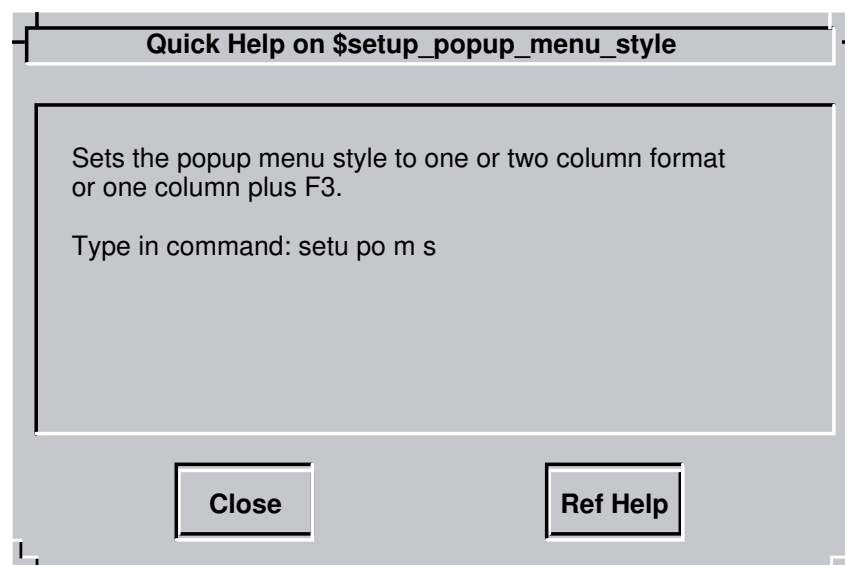


Figure 2-15. Quick Help Message Box

Reference help can be accessed in these ways:

- From **Help > On Commands** in menu bar: to get listing of available commands.
- From **Help > On Functions** in menu bar: to get a prompt bar in which you enter the name of the function on which you want help.

The BOLD Browser is then invoked automatically, and the function page in the appropriate reference manual describing the specified function is displayed.

- From a Quick Help message box: select the Ref Help button.

Online Documentation

Board Station manuals are available online. Most can be accessed through the BOLD Browser from the **File > Open > Document** or **File > Open > Bookcase** menu items. The Board Station manuals are listed in the following bookcases and as individual documents: MCM Station, PCB, and PCB Interfaces.

The bookcase is a convenient listing of all manuals associated with a Mentor Graphics product. Table 2-2 lists the documentation bookcases available online. Figure 2-16 shows the BOLD Browser navigator listing the PCB bookcase.

Table 2-2. Documentation Bookcases

Analog Simulation	Logical Cable and Physical Cable
Bold Online Information	Manufacturing Advisor
Customer Support Bulletins	Miscellaneous
Design Creation	Print Servers
Digital Component Libraries	Quick Reference Booklets
Digital Modeling	Release Notes
Digital Simulation	Synthesis
Falcon Framework	System Management
General	System-1076
Hybrid Station	Training Workbooks
IC Station	Transition
Library Management	

Not all Board Station documents are available from the Bold Browser. Board Station documents that are not available from the Bold Browser are available to view with FrameViewer or FrameMaker. These documents are located in the `$MGC_HOME/shared/pkgs/pcb_docs` directory. This directory must be installed by your system administrator. If you cannot find this directory, ask your system administrator to install it for you.

For the FrameMaker/FrameViewer library, the FrameMaker document *Overview of PCB FrameMaker Documents* describes accessing the manuals and defines the PCB manuals available in FrameMaker/FrameViewer. This document is located at `$MGC_HOME/shared/pkgs/pcb_docs/overview`.

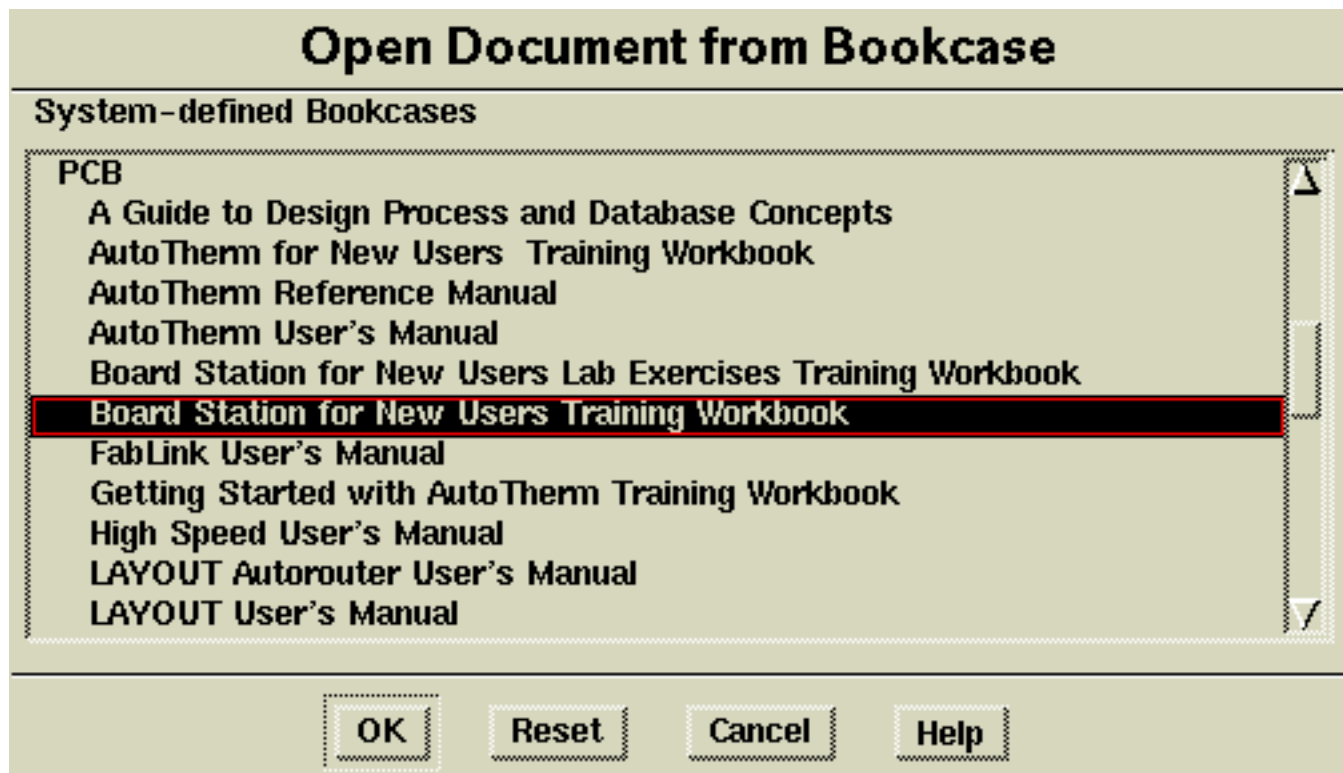


Figure 2-16. PCB Bookcase

Lab Exercise

This lab exercise familiarizes you with the user interface. You explore the PCB product documentation with the BOLD Browser tool. The BOLD Browser gives you access to Mentor Graphics online product documentation.

While you explore with the BOLD Browser, you use the various features of the Common User Interface including manipulating windows and filling in dialog boxes.

Upon completion of this lab exercise you should be able to:

- Open documents in the BOLD Browser.
- Manipulate the document windows.
- Make choices in a dialog box.
- Use the prompt bar to select command options.
- Use hyperlinks in a document to find topic items.

Turn to Module 1—Lab 1: "The Environment".

Lab 1

The Environment

Introduction

This lab exercise familiarizes you with the user interface. You explore the PCB product documentation with the BOLD Browser tool. The BOLD Browser gives you access to Mentor Graphics online product documentation.

While you explore with the BOLD Browser, you use the various features of the Common User Interface including manipulating windows and filling in dialog boxes.

Upon completion of this lab exercise you should be able to:

- Open documents in the BOLD Browser.
- Manipulate the document windows.
- Make choices in a dialog box.
- Use the prompt bar to select command options.
- Use hyperlinks in a document to find topic items.

Procedure

In this procedure, you invoke the Design Manager and explore the online documentation using the Bold Browser.

Invoking and Setting up the Design Manager

1. Log onto the workstation.

The first tool that you are going to invoke is the Design Manager. All Mentor Graphics tools can be invoked by the Design Manager. The Design Manager also allows you to perform data manipulations, such as copying designs. In preparation for this lab, you setup the Design Manager session and display a Tools window.

2. Enter the following command exactly as shown here in at a shell prompt to invoke the Design Manager:

`$MGC_HOME/bin/dmgr`

Shell commands are case sensitive. You invoke all PCB tools in this course using the Design Manager.

3. If you are using a HP workstation, enlarge the Design Manager window to fill the display by placing the cursor over the Window Maximize icon in the far upper-right corner of the Design Manager window and clicking the Select mouse button (the left mouse button by default). Refer to Figure 2-17. If you are using a Sun workstation, choose the **Full Size** menu item from the window menu.



Figure 2-17. Location of the Window Maximize Icon

You can use the Window Maximize icon to toggle the Design Manager session window (or any other application session window) between full size and the window's original size by placing the cursor on the icon and clicking the Select mouse button.

If the session window is full size, and you want to return the Design Manager window to its original size, you click the Select mouse button again on the same icon. Leave the window full size for now.

In the next step, you will perform a session setup.

4. Choose the **Setup > Default Editor...** menu item. To choose the menu item, place the cursor on **Setup** in the menu bar and hold down the Select mouse button. When the pulldown menu is displayed, move the cursor down the menu to the **Default Editor...** menu item, and release the Select mouse button.

From now on, you will not be directed how to choose a menu item.

The Setup menu allows you to specify a default file editor and the positioning of icons in the Display Manager's windows.

The Design Manager Session Setup dialog box asks for a default ASCII file editor. The default editor is the Notepad.

5. If the Notepad editor is not highlighted, choose the Notepad editor by clicking the Select mouse button on the button above the text *Notepad*. Press the **OK** button in the dialog box.
6. If a dialog box appears asking, *Save setup changes in default startup file?* press the **Yes** button.

You might also see another dialog box stating that a default startup file already exists. If you see this dialog box, choose **Yes** to overwrite it if you want to change your default startup for the Design Manager.

Next, you set the spacing between icons.

7. Choose the **Setup > Iconic Layout...** menu item.

The rectangles, labeled *A* and *B*, represent the horizontal and vertical spacing between icons. You can move the *B* icon by placing the cursor on the rectangle, pressing the Select mouse button, and dragging the rectangle.

8. Drag both *B* rectangles to increase the space between the icons shown.

This action makes it easier to read the names of the icons.

9. If you want to see the icons staggered in the displayed window, press the **Yes** button next to the question: Stagger alternate icons?
10. When you are finished with the Iconic Window Layout Setup dialog box, press the **OK** button. Another dialog box, titled *Save Setup Changes in Default Startup File* might be displayed, if it is, choose **Yes**.

You might also see another dialog box stating that a default startup file already exists. If you see this dialog box, choose **Yes** to overwrite it if you want to change your default startup for the Design Manager.

11. If the tools window is not displayed (showing icons of all the tools), choose the **Windows > Open Tools Window** menu item.

To invoke Mentor Graphics applications from the Design Manager, you need a Tools window. The Tools window shows icons of all the Mentor Graphics applications that you can access.

Invoking the BOLD Browser

1. Find the BOLD Browser icon in the Tools window. Look for an icon showing an open 3-ring binder with a bookmark. Invoke the BOLD Browser from the Tools window by placing the cursor on the BOLD Browser icon and *quickly* double clicking the Select mouse button.

The icon changes to a reverse image, indicating you have selected it, and the cursor changes to an hour glass, indicating the tool for the selected icon is being invoked. If the cursor did not change to an hour glass, you should try double clicking on the icon again. You need to perform the double click *very* quickly. Next, a dialog box appears requesting more information.

2. Complete the dialog box as follows, then press **OK** to complete the dialog box.

If you know which document you want to open, you can specify the name here: [Leave this box blank]

Do you want the BOLD Browser to service requests to display online help? **Yes**

Do you want documents to display color? **Yes**

A new window is created in which the BOLD Browser is automatically invoked.

You did not specify a document in the dialog box, because it is easier to choose a document from a list in a dialog box later. You set a switch that causes the BOLD Browser to respond to requests for reference help. If you ask for reference help in FabLink or any other Board Station tool, the BOLD Browser intercepts the request and locates the reference pages. You also requested the BOLD Browser to display the text and illustrations of a document in color. Many of the illustrations in the Board Station documentation set are easier to understand if you view them in color.

3. Spend a few moments familiarizing yourself with the mouse and the look of the BOLD Browser.

The functionality that you see in the Common User Interface of the BOLD Browser is similar to that of other Mentor Graphics tools.

Opening a Document

1. In the BOLD Browser enlarge the BOLD Browser window to fill the display.

If you are working on an HP workstation, you can click the Select mouse button on the Maximize Window icon. On Sun workstations, choose the **Full Size** menu item from the window menu.

It is easier for you to read the documents you open if the window fills the display.

2. Choose the **File > Open > Bookcase...** menu item.

A list of documentation sets available to the Browser displays in a dialog box.

3. Move the cursor to the list of document bookcases. Use the scroll bar at the right edge of the dialog box to scroll through the list of bookcases until you find the **PCB** bookcase.
4. Select the **PCB** bookcase in the list by placing the cursor on it and clicking the Select mouse button. Next, press the **OK** button at the bottom of the dialog box.

The entry expands to show a list of all PCB product documentation.

5. Examine the list of PCB product documentation. Scroll down the list until you find **Using PCB Tools**. Select the name of the document and then **OK** the dialog box.

The dialog box is removed, and a subwindow appears showing the title page of the document.

6. Make the document subwindow full size in the BOLD Browser session window by placing the cursor on the Window Maximize icon at the upper-right corner of the document subwindow and clicking the Select mouse button.

Another way to change the size of a document subwindow is to place the cursor on the window border, hold down the Select mouse button, drag the window border to the size you want, and release the Select mouse button.

Using Dialog Boxes

This section uses a dialog box to change the session settings.

1. In the BOLD Browser, choose the **MGC > Setup > Session...** menu item.

The Session Setup dialog box is displayed.

2. Move the cursor to the check buttons at the upper-right corner of the dialog box. Place the cursor on the check button labeled **Show Menu Bar** and click the Select mouse button to unselect the button. Notice that the unselected check button now appears raised and the same color as the dialog box background.
3. **OK** the dialog box. Note that the menu bar is no longer visible.

In the next step, you will restore the session. To do so, you need the Session Setup dialog box. Where is the MGC menu? You turned it off with the Menu Bar. There is another area where the setup menu is available.

4. If your document subwindow is maximized, reduce the size so some of the BOLD Browser session window background shows. Place the cursor in the session window background outside the document subwindow. Press the Menu mouse button (the right mouse button, by default) and choose the **[Session] Setup > Session** menu item. Restore the Menu Bar using the Session Setup dialog box.

Using Prompt Bars

Prompt bars allow you to supply missing arguments. To test prompt bars, use the **goto page** command.

1. Make sure the document window is active. If the document window banner is not highlighted (active) in blue (default color), place the cursor in the document window and click the Stroke mouse button (the center mouse button by default).

Commands you enter, and menu items you choose apply to the active window. You must make the window you want acted upon active before executing a command or menu item.

2. With the cursor anywhere within the document window, type **goto page** or **got pa**.

As you begin typing, a popup command line displays.

3. When you are finished entering the **goto page** command, press the Return key.

This application command calls a builtin function `$$goto_page_form()`, which displays the prompt bar for the **goto page** command.

The prompt bar has an entry box to type in a page number plus a number of fields with stepper buttons that you use to make other choices. Pressing the Tab key causes the cursor to jump to the next field in the prompt bar.

4. Press the Tab key several times until the active entry location returns to the page number prompt.

You can press the Shift-Tab key to move the cursor one prompt field to the left.

5. Type **3-12** in the box, and execute the prompt bar by either clicking on the **OK** button or by pressing the Return key.

6. Try another page number by typing **goto page** in the popup command line and entering a new page number in the prompt bar.

Do not be afraid to use any page number. If the page does not exist, a message appears in the message area. Try another page number if the first does not work. You can also go to the Table of Contents to see what page numbers exist. Remember that every application command is linked to an AMPLE function. The AMPLE function from which **goto page** is derived is \$goto_page().

Using the BOLD Browser

The remainder of this lab covers the BOLD Browser and the INFORM online documentation. This section includes few procedures. Instead, there is just a task list. These tasks refer to the *PCB PACKAGE User's Manual*. If you have difficulty performing the tasks, open the *BOLD Browser User's Manual* where you can find the procedures for the rest of your tasks in that manual.

For the remainder of this lab, your tasks are:

1. Go to the Table of Contents.

The two icons at the upper-left corner of the document subwindow are the Table of Contents icon and the Index icon. Clicking on the left icon displays the Table of Contents. Clicking on the right icon displays the Index.



The document subwindow popup menu contains a Travel Log menu item. The system automatically keeps a record of the pages you have viewed. To view the previous page, you place the cursor in the document subwindow, and choose the [Document Area] Travel Log > Backward menu item. If you choose the back [Document Area] Travel Log > Summary menu item a dialog box is displayed listing the pages you have previously viewed. You can select any page from the list and OK the dialog box to view any previously viewed page.

2. In the Table of Contents, place the cursor on a subject so you see the cursor change from an arrow to a hand/finger pointer, and click the Select mouse button.

The page with the subject you chose is displayed. The object you clicked on in the Table of Contents is called a Hyperlink. Anytime you see a hyperlink (when the cursor changes from an arrow to the hand/finger pointer), you can click on it to view more information on that subject. You can use the Travel Log to return back to the previous page you viewed.

3. Return to the Table of Contents and view another subject.
4. Use the Travel Log to go back to a previously viewed page.
5. Click on the Index icon in the document window banner, and then in the index click on a hyperlink to view that subject.
6. Activate the Session window and choose the **File > Open > Bookcase** menu item. Choose the PCB bookcase and explore other documents.
7. Modify your Browser display. Try the zoom feature. At what point is it comfortable for you? Try changing the colors of the document background or text.

If you need help changing the display, open the *BOLD Browser User's Manual*.

8. In an active document window, try a text search. Try any search pattern that interests you.

Ending the Lab Exercise

The process for ending each lab exercise is similar. First, you save the design data if appropriate, then you close the application, and finally you close the shell. You then close the Design Manager and its shell. When you finish closing applications and shells, you log out of the workstation.

1. Close the BOLD Browser session by moving the cursor to the Window menu button in the upper-left corner of the session window and pressing the Menu mouse button (the right mouse button, by default). From the menu that appears choose **Close**.
2. If the shell in which the BOLD Browser ran remains, close it.
3. Use exactly the same process to close the Design Manager session and its shell.
4. Log out of the workstation.

Congratulations! You have completed the *Introduction to Board Station* module. The next module of the *Board Station for New Users Training Series* is Module 2: *Preparing a Design for PCB*.

Glossary

This subsection includes definitions of some important new terms. The terms are listed alphabetically. For a complete glossary of Version 8 terms, refer to the *PCB Products Overview Manual*.

AMPLE

Acronym for Advanced Multipurpose Language. AMPLE is a structured, procedural programming language that provides the environment for application and user-defined functions, menus, and dialog boxes.

BOLD

A tool for viewing, annotating, and printing online documentation. Beginning with V8, all Mentor Graphics product documentation is available online and accessed through the BOLD product.

CD ROM

Acronym for Compact Disc - Read Only Memory. CD ROMs are optical disks used as a read-only data storage media. All Mentor Graphics V8 product documentation is available on the INFORM CD ROM.

check button

A control used to select settings that are not mutually exclusive. The visual cue to the selection is frequently that the button is filled in or checked.

clipboard	A device used to store text or graphics during cut-and-paste operations. The Notepad text editor uses a clipboard.
concurrent design	A design methodology in which multiple product parameters are taken into consideration at the same time, through the sharing of analytical information between teams of designers in different disciplines.
container	A special type of design object that can contain other design objects. Containers are the equivalent of file system directories.
control	A graphical device for entering information in a dialog box or prompt bar. Some commonly used controls include entry boxes, push buttons, radio buttons, and list boxes.
conversion	The process of changing pre-V8 data and userware into formats compatible with the V8 tools.
data model	An internal representation of a collection of data, used and maintained by a software program. An example of a data model is the Electronic Design Data Model (EDDM).
design object	A representation of design data as a single entity with associated behavior, rather than as a collection of directories, files, and links with no unique characteristics.
dialog box	A window that appears on the screen when you choose certain menu items. Labels in the dialog box prompt you for the information required to execute the function associated with the menu item.

Falcon Framework	The underlying core technology used and shared by all V8 design tools. The Falcon Framework consists of a user interface based on OSF/Motif guidelines, a design database management system (DDMS), an advanced multipurpose language (AMPLE), online documentation (INFORM), and a text editor (Notepad).
float	The ability to execute a software program from any node in a network rather than from a specific node. The Mentor Graphics Licensing Administrator (MGLA) controls which V8 tools can float. All V8 PCB products are available with floating licenses.
function	A set of instructions designed to run in an application. These instructions can be other functions, commands, or programming constructs recognized by Mentor Graphics applications. Functions are the basic programming units of AMPLE.
icon	A small graphical image used to represent an object such as a design tool, design object, or window. You can turn windows into icons to save space when the windows are not being used.
INFORM	The CD ROM containing the product documentation for Mentor Graphics tools. The INFORM information is accessed with BOLD.
maximize	To enlarge a window to its maximum allowable size.
message box	The generic name for any dialog box that provides information, gives the current state of an action in progress, asks a question, issues a warning, or draws attention to an error.

minimize	To turn a window into an icon.
mnemonic	A single character (usually the initial character) of a menu item that is equivalent to the complete name of the menu item. Pressing the single character on the keyboard while the menu is displayed initiates the selection of the menu item.
Motif	The name of the guidelines defined by the Open Software Foundation (OSF) for user interfaces to software applications. All V8 tools use a common user interface developed from these guidelines.
node-locked	A term that refers to software programs restricted to run on one or more specific nodes. The restriction is enforced by routines in the software that match the ID numbers of the nodes to authorization codes in software.
Notepad	A full-featured text editor that you can invoke through the user interface or through AMPLE functions. Available features include font selection and size, search and replace capabilities, auto-wrap, auto-deletion, and file manipulation. You can select text for deletion, copying, clipboard storage, and cut-and-paste operations. Notepad is programmable, with a set of approximately 100 AMPLE functions.
Open Software Foundation (OSF)	A consortium of computer hardware and software manufacturers formed to develop industry standards for software products.

prompt bar	A single-line dialog box based on the definition of an AMPLE function. The prompt bar contains the function name, a field for each required argument of the function, and a set of buttons consisting of an Options button, a Cancel button, and an OK button.
push button	A control that simulates an actual push button. Click the Select mouse button to select (push) the button and start an action.
radio button	A control used in a dialog box that operates like the button of a car radio. Each radio button represents a mutually exclusive selection. The selection of a radio button typically sets a state or mode.
session window	The bounding window for a V8 tool session. Everything associated with a tool session appears within a Session window. A Session window is created out of a standard operating system process window when you invoke a tool, and may be moved, resized, and stacked along with other operating system display objects.
subwindow	A window that appears within the boundaries of its parent window. Tool subwindows appear within the session window frame, for example.
userware	Software written in AMPLE that you use to implement or modify the user interface and other programs. Userware includes menu definitions, functions, and macros. All PCB products are provided with default userware.
window frame	The area surrounding a window. A window frame can consist of a border, a window menu button, a title bar, and window control buttons.

INDEX

A

AMPLE
defined G-1

B

BOLD
defined G-1

C

CD ROM
defined G-1
Check button
defined G-1
Clipboard
defined G-2
Concurrent design
defined G-2
Container
defined G-2
Controls
defined G-2
in dialog boxes 2-10
Conversion
defined G-2

D

Data models
defined G-2
Design Manager
invoking xi
Design objects
defined G-2
Dialog boxes
controls 2-10
defined G-2

F

Falcon Framework
defined G-3
Float
defined G-3
Functions
defined G-3

I

Icon
defined G-3
INFORM
defined G-3
Invoke
Design Manager xi

M

Maximize
defined G-3
Message boxes
defined G-3
Mid-Command Freedom 2-15
Minimize
defined G-4
Mnemonics
defined G-4
Motif
defined G-4

N

Node-locked
defined G-4
Notepad
defined G-4

INDEX [continued]

O

OSF
 defined G-4

P

Prompt bars 2-14
 defined G-5
Push buttons
 defined G-5

R

Radio buttons
 defined G-5

S

Session window
 defined G-5
Setup > Session 2-33
Subwindow
 defined G-5

U

Userware
 defined G-5

W

Window frame
 defined G-5

欢 迎 光 临

微波 EDA

<http://www.mweda.com>

EDA学习网

<http://edastudy.ik8.com>

<http://www.mweda.com>

<http://edastudy.ik8.com>

本站专业提供各种微波仿真软件和 PCB 设计软件的安装光盘和学习培训教程，本站提供的所有教程教材都是公司培训用的，很系统的讲述了相关软件的应用。主要项目有：

破解软件类：微波仿真软件

ADS2004A	ADS2003C	ADS2003A		
HFSS9.2	HFSS9.1	HFSS9.0	HFSS8.0	
Ansoft Designer1.1	Ansoft Serenade8.71	Ansoft Maxwell 10	Ansoft SIWave	
Microwave Office2002	Sonnet Suite Pro 9.52	CST5.0		
Super NEC 2.5	Zeland IE3D9.2	XFDTD 6.0		

破解软件类：PCB 工具软件

Mentor EN2004	Mentor EN2002	Mentor ePD2004	Mentor SDD2004
Mentor WG2004	Mentor WG2002	Mentor ISD2004	
PowerPCB5.0	PowerLogic5.0	Orcad10.3	
PADS2005	PADS2004		
Cadence SPB15.2 (Allegro 15.2)		Cadence PSD15.0 (Allegro 15.0)	

软件学习、培训教程：

Mentor EN: Mentor EN 原版培训教程
Mentor EN 视频教程

Mentor WG: Mentor Expedition / Mentor WG 中文用户手册
Mentor Expedition (WG) PCB Training Workbook
Mentor DxDesigner Design Processing Training Workbook

HFSS : HFSS 9.2 入门与提高教程
HFSS 9.0 入门与提高教程
HFSS9 视频教程
HFSS9 Training Tutorials
HFSS8 Training Manual
电子科大 HFSS8 教程

ADS: ADS 中文基础教程
ADS 设计实验教程
ADS 入门教程
ADS2003 Training Workbook: ADS2003 fundamental
ADS Training Workbook for Momentum
ADS Customization Training Workbook

Using ADS Communication Systems Designer

Using ADS to Design WCDMA/3GPP Communication Systems

Ansoft Designer:

Ansoft Designer 2.1 Full Book

Ansoft Designer System Training Guide

Ansoft Designer 1.1 入门与提高

Ansoft Designer 多媒体教程

Ansoft Designer Training 讲义

Cadence Allegro:

Allegro 15.2 原版教程

Concept-HDL 15.2 原版教程

Allegro 14.2 原版教程

Allegro 视频教程

Allegro PCB Layout 高速电路板设计

注：本站还有Ansoft Maxwell, Ansoft Serenade, Ansoft Q3D , Zeland IE3D, PowerPCB ,
Orcad等的培训教程。详情可登陆：<http://www.mweda.com> 或 <http://edastudy.ik8.com>

欢 迎 光 临

微波 EDA

<http://www.mweda.com>

EDA学习网

<http://edastudy.ik8.com>

<http://www.mweda.com>

<http://edastudy.ik8.com>

本站专业提供各种微波仿真软件和 PCB 设计软件的安装光盘和学习培训教程，本站提供的所有教程教材都是公司培训用的，很系统的讲述了相关软件的应用。主要项目有：

破解软件类：微波仿真软件

ADS2004A	ADS2003C	ADS2003A		
HFSS9.2	HFSS9.1	HFSS9.0	HFSS8.0	
Ansoft Designer1.1	Ansoft Serenade8.71	Ansoft Maxwell 10	Ansoft SIWave	
Microwave Office2002	Sonnet Suite Pro 9.52	CST5.0		
Super NEC 2.5	Zeland IE3D9.2	XFDTD 6.0		

破解软件类：PCB 工具软件

Mentor EN2004	Mentor EN2002	Mentor ePD2004	Mentor SDD2004
Mentor WG2004	Mentor WG2002	Mentor ISD2004	
PowerPCB5.0	PowerLogic5.0	Orcad10.3	
PADS2005	PADS2004		
Cadence SPB15.2 (Allegro 15.2)		Cadence PSD15.0 (Allegro 15.0)	

软件学习、培训教程：

Mentor EN: Mentor EN 原版培训教程
Mentor EN 视频教程

Mentor WG: Mentor Expedition / Mentor WG 中文用户手册
Mentor Expedition (WG) PCB Training Workbook
Mentor DxDesigner Design Processing Training Workbook

HFSS : HFSS 9.2 入门与提高教程
HFSS 9.0 入门与提高教程
HFSS9 视频教程
HFSS9 Training Tutorials
HFSS8 Training Manual
电子科大 HFSS8 教程

ADS: ADS 中文基础教程
ADS 设计实验教程
ADS 入门教程
ADS2003 Training Workbook: ADS2003 fundamental
ADS Training Workbook for Momentum
ADS Customization Training Workbook

Using ADS Communication Systems Designer

Using ADS to Design WCDMA/3GPP Communication Systems

Ansoft Designer:

Ansoft Designer 2.1 Full Book

Ansoft Designer System Training Guide

Ansoft Designer 1.1 入门与提高

Ansoft Designer 多媒体教程

Ansoft Designer Training 讲义

Cadence Allegro:

Allegro 15.2 原版教程

Concept-HDL 15.2 原版教程

Allegro 14.2 原版教程

Allegro 视频教程

Allegro PCB Layout 高速电路板设计

注：本站还有Ansoft Maxwell, Ansoft Serenade, Ansoft Q3D , Zeland IE3D, PowerPCB , Orcad等的培训教程。以及射频/微波/天线类英文原版书籍。详情可登陆：

<http://www.mweda.com> 或 <http://edastudy.ik8.com>